

Purdue University
Purdue e-Pubs

International Compressor Engineering Conference

School of Mechanical Engineering

2010

Flow Simulation Through Moving Hermetic Compressor Valves Using the Immersed Boundary Method

Jose Luiz Gasche
Unesp-Univ Estadual Paulista

Tadeu Tonheiro Rodrigues
Unesp-Univ Estadual Paulista

Julio Militzer
Dalhousie University

Follow this and additional works at: <https://docs.lib.purdue.edu/icec>

Gasche, Jose Luiz; Rodrigues, Tadeu Tonheiro; and Militzer, Julio, "Flow Simulation Through Moving Hermetic Compressor Valves Using the Immersed Boundary Method" (2010). *International Compressor Engineering Conference*. Paper 1999.
<https://docs.lib.purdue.edu/icec/1999>

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact epubs@purdue.edu for additional information.

Complete proceedings may be acquired in print and on CD-ROM directly from the Ray W. Herrick Laboratories at <https://engineering.purdue.edu/Herrick/Events/orderlit.html>

Flow Simulation Through Moving Hermetic Compressor Valves Using the Immersed Boundary Method

Tadeu Tonheiro RODRIGUES¹, José Luiz GASCHE^{1*}, Júlio MILITZER²

¹Unesp-Ilha Solteira, Department of Mechanical Engineering,
Ilha Solteira, São Paulo, Brazil
(55 18 3743-1035, tadeu.tonheiro@gmail.com, gasche@dem.feis.unesp.br*)

²Dalhousie University, Mechanical Engineering Department,
Halifax, Nova Scotia, Canada
(1 902 494-3947, jmilitze@dal.ca)

ABSTRACT

In this work the flow through a radial diffuser representing the valve system of a hermetic compressor is solved numerically by using the Immersed Boundary Method with the Virtual Physical Model, which has been successfully used for numerical simulation of flows through complex geometries, including fluid-structure interaction. The reed movement was imposed by prescribing a constant velocity to the reed, with the displacement of the reed limited through a fixed gap between the reed and the inferior disk. The performance of the method was evaluated for Reynolds number of 1500. The results showed interesting flow patterns not identified in static simulations of the flow. The most important conclusion of the work is that the methodology showed to be suitable for studying the flow through the radial diffuser with reed movement, indicating that it can be used in the future for the real geometry of the valve system, including the fluid-structure interaction between the reed and the flow.

1. INTRODUCTION

The valve is one of the main components of a hermetic compressor, given it controls the mass flux inside the compressor. The opening and closing movements are governed by the pressure difference applied by the refrigerant flow over the reed. Therefore, it is essential to fully understand the flow through the valve in order to improve the valve design and to enhance the overall efficiency of the compressor.

Several studies have been made to characterize the main features of the flow through the valve. Numerical solutions for incompressible laminar flows have been obtained by Hayashi *et al.* (1975), Raal (1978), Piechna and Meier (1986), Ferreira *et al.* (1989), Gasche *et al.* (1992) and Possamai *et al.* (2001). Numerical solutions for incompressible turbulent flows have been obtained by Deschamps *et al.* (1996) and Colaciti *et al.* (2007). Experimental work on this subject has been performed by Wark and Foss (1984), Ferreira and Driessen (1986), Tabakabai and Pollard (1987), Ervin *et al.* (1989) and Gasche *et al.* (1992). Nevertheless, most of these studies are limited to static analysis of the problem, disregarding the movement of the valve reed and flow-structure interaction. However, a few studies in this area, Matos *et al.* (2000) and Matos *et al.* (2001), do not use a static analysis.

One of the main challenges for modeling the problem is the complex geometry of the valve, and for this reason simpler geometries have been adopted, specially the radial diffuser. Despite the valve geometry, the treatment of valve reed movement is challenging for the current computational fluid dynamics methods. The use of body-fitted meshes, where the computational mesh is set to fit to the body, introduces several computational penalties once that, for each displacement of the valve reed, the mesh must be updated for the discretization of the new computational domain. This procedure is quite difficult and requires extensive computational resources.

The main purpose of the present work is to apply an alternative approach for dealing with the valve reed displacement, namely, the Immersed Boundary Method developed by Peskin (1972). The Immersed Boundary Method has been successfully used for different applications, especially for problems involving fluid-structure interaction.

The Immersed Boundary Method uses a simple fixed grid (Eulerian grid) for solving the flow equations and models the presence of an interface using a moving Lagrangian grid with the addition of a force field in the momentum equations for the cells located around the Lagrangian points.

In this work the flow through a radial diffuser representing the valve system is solved numerically by using the Immersed Boundary Method with the Virtual Physical Model (Lima e Silva *et al.*, 2003), used for calculating the force field. The radial diffuser is simulated using a two-dimensional approach using the cylindrical coordinate system. The valve reed is moved artificially by imposing a fixed velocity for the reed and a limited range for the reed displacement. The results were obtained for Reynolds number of 1500 and shows interesting flow patterns not identified in the static simulation of the flow. The most important conclusion of the work is that the methodology showed to be suitable for studying the flow through the radial diffuser with reed movement, indicating that it can be used in the future for the real geometry of the valve system, including the fluid-structure interaction between the reed and the flow.

2. MATHEMATICAL FORMULATION

In this study a two-dimensional, unsteady, incompressible, and isothermal flow of a Newtonian fluid in cylindrical coordinates was considered to solve the flow through the radial diffuser. The governing equations are the mass conservation and the momentum equations written in the r direction (radial) and x direction (axial), given by:

$$\frac{1}{r} \frac{\partial(r\rho u)}{\partial r} + \frac{\partial(\rho w)}{\partial z} = 0 \quad (1)$$

$$\rho \left(\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial r} + v \frac{\partial v}{\partial x} \right) = - \frac{\partial p}{\partial r} + \mu \left(\frac{\partial^2 v}{\partial r^2} + \frac{1}{r} \frac{\partial v}{\partial r} - \frac{v}{r^2} + \frac{\partial^2 v}{\partial x^2} \right) + F_r \quad (2)$$

$$\rho \left(\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial r} + v \frac{\partial u}{\partial x} \right) = - \frac{\partial p}{\partial x} + \mu \left(\frac{\partial^2 u}{\partial r^2} + \frac{1}{r} \frac{\partial u}{\partial r} + \frac{\partial^2 u}{\partial x^2} \right) + F_x \quad (3)$$

where u and v are the velocity components in the axial and radial directions, respectively; p is the pressure; and ρ and μ are the density and the viscosity, respectively. The terms F_r and F_x are Eulerian force density fields, which models the immersed boundary, being thus responsible for the representation of the body inside the flow. The Eulerian forces are calculated through the distribution of the Lagrangian interfacial forces, f_i , with the following equation:

$$F_i(\vec{x}_{ij}) = \sum_k D_{ij} \left(\left\| \vec{x}_{ij} - \vec{x}_k \right\| \right) f_i(\vec{x}_k) \frac{\Delta V_k}{\Delta V_{ij}} \quad (4)$$

where ΔV_k is the Lagrangian point volume, ΔV_{ij} the Eulerian point volume, \vec{x}_k the Lagrangian point position, \vec{x}_{ij} the Eulerian point position, and D_{ij} is the distribution function having Gaussian function properties.

Several models to calculate the Lagrangian interfacial force, f_i , have been developed (Mittal and Iaccarino, 2005). In this work, the Virtual Physical Model proposed by Lima e Silva *et al.* (2003) is used to calculate the interfacial Lagrangian force, which is defined as:

$$\vec{f}(\vec{x}_k, t) = \underbrace{\rho \frac{\partial \vec{V}(\vec{x}_k, t)}{\partial t}}_{\vec{f}_a} + \underbrace{\rho (\vec{V} \cdot \vec{\nabla}) \vec{V}(\vec{x}_k, t)}_{\vec{f}_l} - \underbrace{\mu \nabla^2 \vec{V}(\vec{x}_k, t)}_{\vec{f}_v} + \underbrace{\vec{\nabla} p(\vec{x}_k, t)}_{\vec{f}_p} \quad (5)$$

where \vec{f}_a , \vec{f}_i , \vec{f}_v , e \vec{f}_p represent, respectively, the inertial acceleration, viscous and pressure forces (by unit volume) acting on the fluid particle at the interface. The terms of Equation (5) are calculated using the Eulerian field data. Details of the implementation are founded in Lima e Silva *et al.* (2003).

3. NUMERICAL METHOD

The Finite Volume Method is used to discretize the governing equations, using a staggered grid. The Power-law scheme is used for interpolating the convective-diffusive terms and the SIMPLEC algorithm is used to treat the pressure-velocity coupling. The discretization of the transient term is accomplished by using the three-time level scheme, a fully implicit second order scheme. The algebraic equation systems are solved iteratively using the TDMA algorithm. Figure 1 shows the computational domain idealized for the valve simulation, the boundary conditions, and also the geometric parameters of the valve.

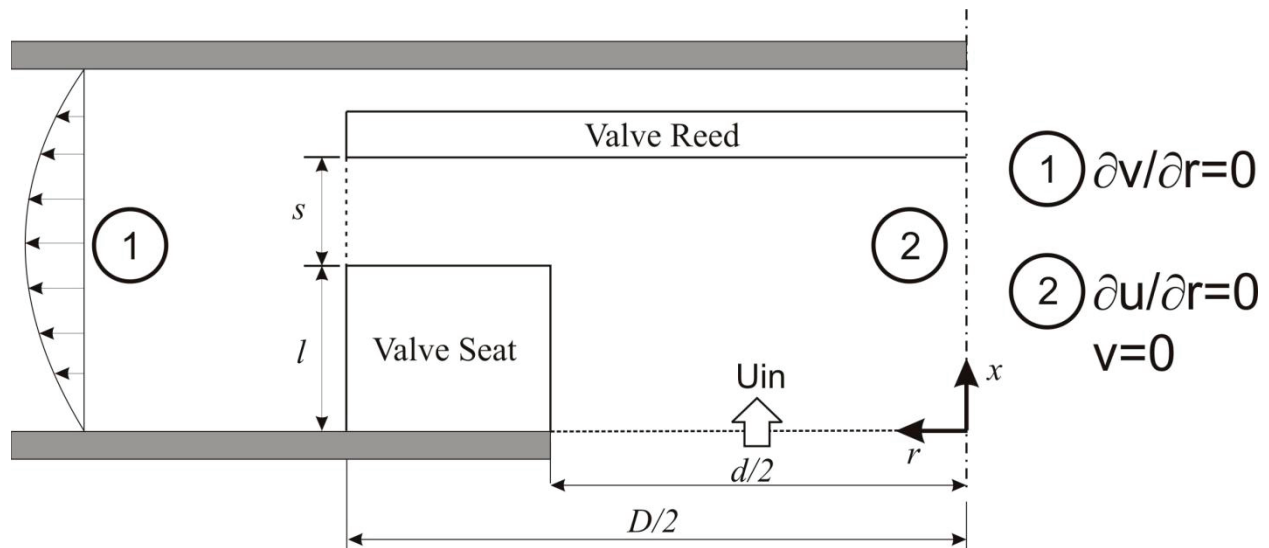


Figure 1: Schematic diagram of the computation domain and boundary conditions for the valve simulation.

In this work, the following geometric parameters are considered: the valve orifice diameter is $d=3$ cm and the diameter relation is equal to $D/d = 1.5$; the height of the valve seat is $l = 0.5$ cm; the gap variation between the valve seat and the valve reed (s/d) varies in the range of $0.07 < s/d < 0.10$; the dimensions of the computational domain are 1 cm and 2.75 cm in the directions x and r , respectively.

The mesh used to perform the simulation is showed in Figure 2. The region with uniform mesh configuration (shadow region) is the region where the valve reed moves. Due to the form of the adopted distribution function, D_{ij} , the mesh where the Eulerian force is applied must be uniform. The mesh refinement in the shadow area is set with 66×306 volumes, totalizing 20196 elements. The remained domain is discretized with a non-uniform mesh, and the total number of elements used for discretizing the entire domain is 31434. The valve seat is modeled by setting the viscosity to infinity.

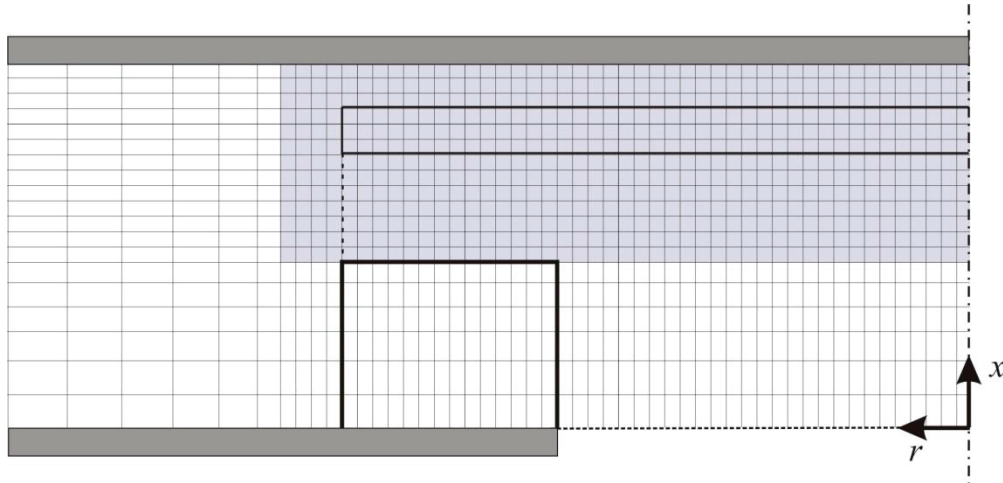


Figure 2: The mesh refinement employed.

4. RESULTS AND DISCUSSION

The simulation was performed by imposing a constant velocity for the valve reed. Both opening and closing movements are set with a constant velocity of $U_{\text{reed}}=3.0$ cm/s. The Reynolds number simulated is 1500, and is defined as follows:

$$Re = \frac{\rho U_{\text{in}} d}{\mu} \quad (6)$$

where U_{in} is the uniform velocity at the inlet of the flow. The results for the dimensionless pressure profiles, P^* , acting on the reed are showed in Figures 3 and 4 for opening and closing movements, respectively, for several s/d relations. The dimensionless pressure is defined by Equation (7).

$$P^* = \frac{p}{\frac{1}{2} \rho U_{\text{in}}^2} \quad (7)$$

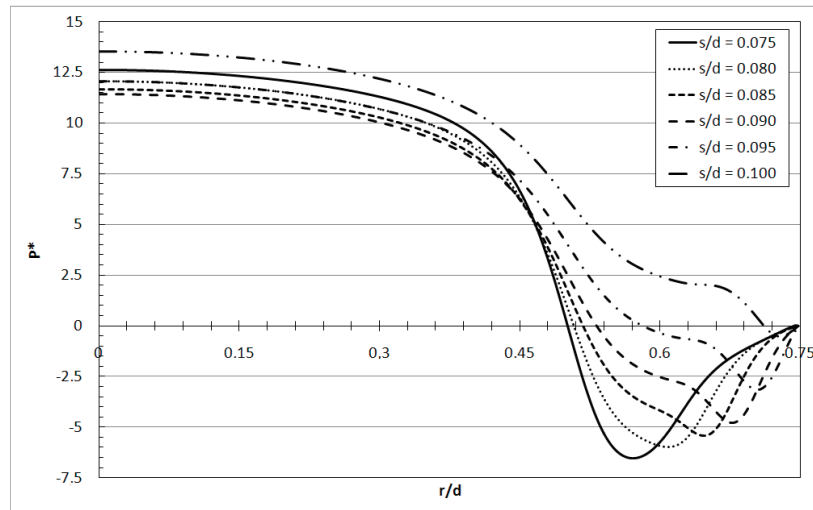


Figure 3: Dimensionless pressure profiles on the reed for opening movement of the valve.

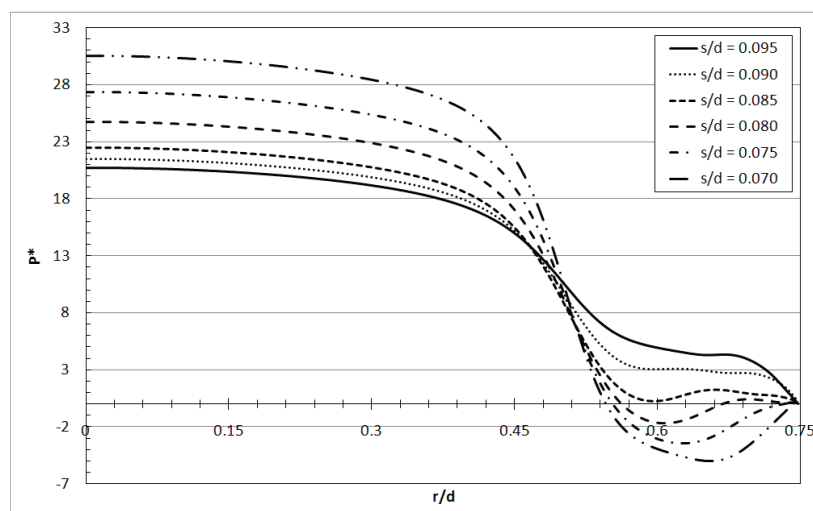


Figure 4: Dimensionless pressure profiles on the reed for closing movement of the valve.

The dimensionless pressure profiles describe a characteristic behavior for all valve reed positions. At the inlet diffuser region there is a large pressure variation, especially for high Reynolds numbers. In this region there is a high fluid acceleration, which produces a high pressure gradient. Downstream this region, the pressure can increase again due to the increasing diffuser cross section area. At the valve orifice region, for $0 \leq r/d \leq 0.4$, there is a practically uniform pressure distribution because this is a stagnation region.

For the opening movement (Figure 3), it is observed that the minimum pressure increases with increasing s/d and is displaced positively in the r direction. It is also noticed a small variation of the pressure at the stagnation point, which decreases with the beginning of the opening movement until $s/d = 0.090$, and increases again for $s/d = 0.100$.

The closing movement (Figure 4) of the valve produces a pressure increase at the stagnation point, but a decreasing of the pressure value for the region where $0.45 < r/d < 0.75$. The pressure values at the stagnation point observed in the closing movement are much higher than those observed for the opening movement, because the relative velocity of the flow is also higher.

Figures 5 and 6 show the stream lines for part of the diffuser region, where the most relevant flow patterns are observed, for the opening and closing movement, respectively.

It is observed in Figure 5 (opening movement) that the height of the main flow recirculation over the valve seat increases for increasing gaps. It is also observed the development of a secondary recirculation on the valve reed surface, which is displaced to the end of the reed for increasing gaps between the valve seat and the valve reed. The development and displacement of the secondary recirculation are responsible for the behavior described for the dimensionless pressure profiles, where the point of minimum pressure is displaced with increasing gaps.

In the closing movement (Figure 6), traces of the secondary recirculation at the end of the reed can be noted. The main characteristics of the flow pattern developed during the movement are the squeezing and stretching process of the recirculation. This process leads to a rupture of the recirculation, resulting in two similar recirculations on the valve seat.

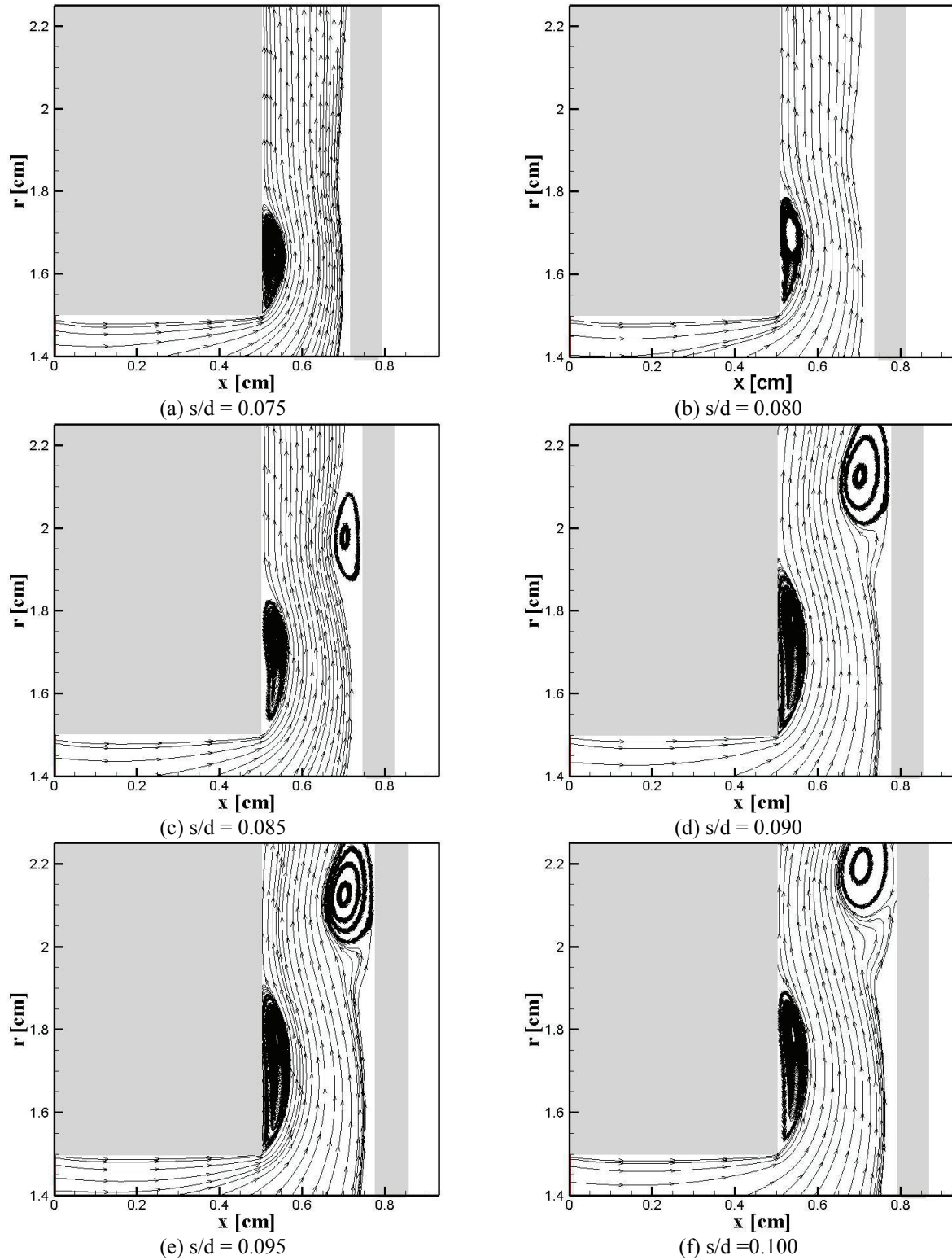


Figure 5: Stream lines for opening movement of the reed.

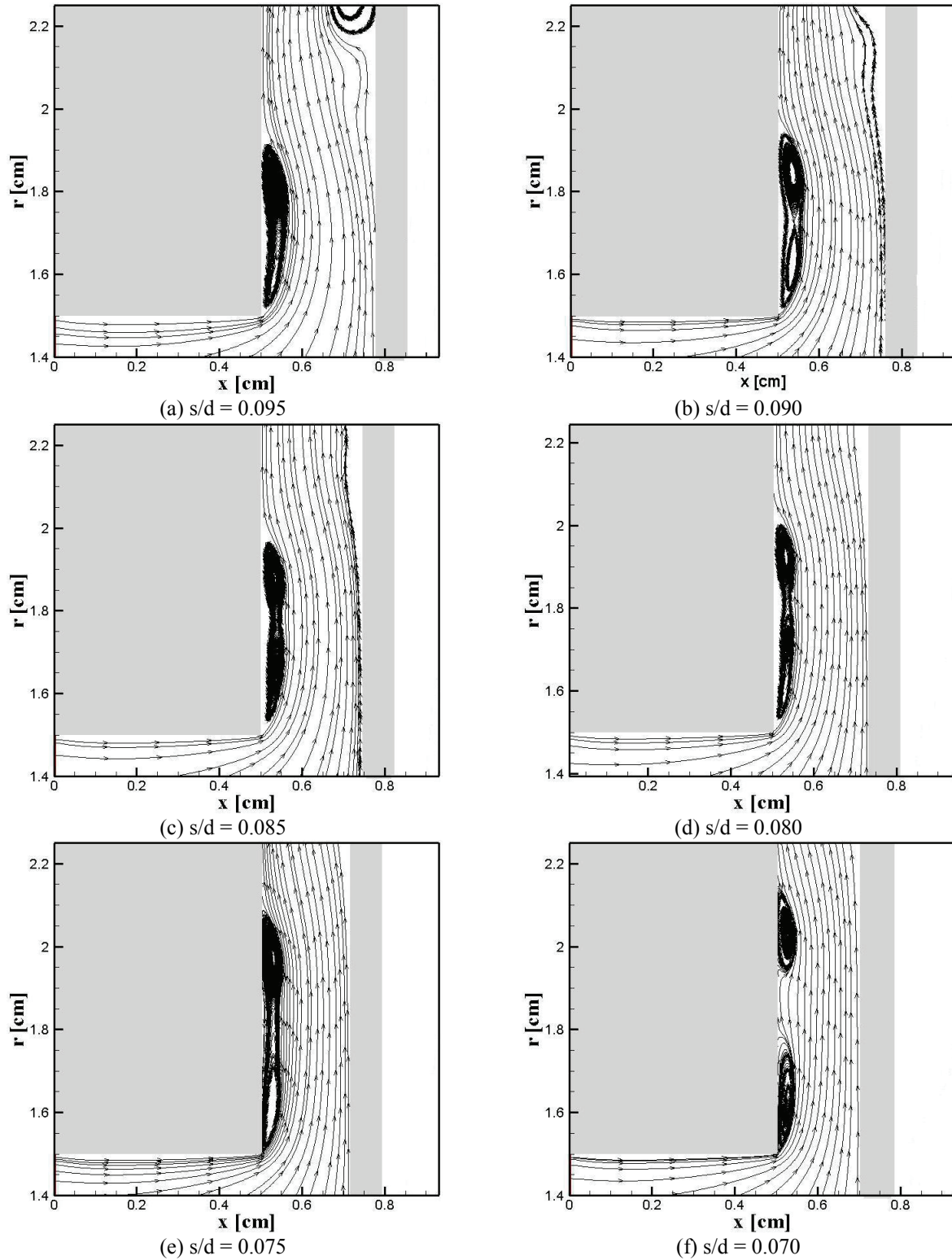


Figure 6: Stream lines for closing movement of the reed.

The flow patterns described were not observed for simulations performed for static reeds, and show a complex correlation of the reed movement with the developing of secondary recirculations.

5. CONCLUSION

This work presented the results obtained for the flow through a radial diffuser, representing the valve system of hermetic compressors, using as alternative approach the Immersed Boundary Method with the Virtual Physical Model. Complex flow patterns were observed by considering the artificial movement imposed to the valve reed, which were not identified in studies for static reed. It was noticed the development of secondary recirculation on the reed surface, which were displaced in the flow direction to the end of the reed as the valve open. During the closing of the valve, it was observed the squeezing and separation of the recirculation formed on the seat surface, giving origin to two smaller recirculations. The presence of recirculation zones introduces flow losses and also can modify the pressure field, which can interfere on the valve performance. The present work was the first attempt of employing the Immersed Boundary Method for modeling the reed movement, indicating that it can be used in the future for predicting the valve working for actual geometries of the valve system, including the fluid-structure interaction between the reed and the flow.

REFERENCES

- Colaciti, A. K, López, L. M. V., Navarro, H. A., Cabezas-Gómez, L., 2007, Numerical Simulation of a Radial Diffuser Turbulent Airflow, *Applied Mathematics and Computation*, vol. 189, p. 1491-1504.
- Deschamps, C.J., Ferreira, R.T.S. e Prata, A.T., 1996, Turbulent Flow Through Reed Type Valves of Reciprocating Compressors, ASME, *International Mechanical Engineering Congress*, Atlanta, EUA.
- Ervin, J.S., Suryanarayana, N.V. e Chai NG, H., 1989, Radial, Turbulent Flow of a Fluid Between Two Coaxial Disks, *Journal of Fluid Engineering*, vol. 111, p. 378-383.
- Ferreira, R.T.S., Driessen, J.L., 1986, Analysis of the Influence of Valve Geometric Parameters on the Effective Flow and Force Areas, *International Compressor Engineering Conference at Purdue*, p. 632-646.
- Ferreira, R.T.S., Prata, A.T., Deschamps, C.J., (1989), Pressure Distribution Along Valve Reeds of Hermetic Compressors, *Experimental Thermal and Fluid Science*, p. 201-207.
- Gasche, J.L., Ferreira, R.T.S., Prata, A.T., 1992, Pressure distribution along eccentric circular valve reed of hermetic compressors, *12th Purdue Int. Compressor Technology Conference*, West Lafayette, USA, pp. 1189-1198.
- Hayashi, S., Matsui, T., Ito, T., 1975, Study of Flow and Thrust in Nozzle-Flapper Valves, *Journal of Fluids Engineering*, vol. 97, p. 39-50.
- Lima e Silva, A.L.F., Silveira-Neto, A., Damasceno, J.J.R., 2003, Numerical simulation of two-dimensional flows over a circular cylinder using the immersed boundary method, *Journal of Computational Physics*, Vol. 189, pp. 351-370.
- Matos, F.F.S., Prata, A.T., Deschamps, C.J., 2000, A Numerical Methodology of Valve Dynamics, *International Compressor Engineering Conference at Purdue*, p. 383-390.
- Matos, F.F.S., Prata, A.T., Deschamps, C.J., 2001, Modeling of Dynamics of Reed Type Valves, *International Conference on Compressor and Coolants*, Slovak, p. 24-31.
- Mittal, R.; Iaccarino, G., 2005, Immersed Boundary Methods, *Annual Review of Fluid Mechanics*, Vol. 37, pp. 239-261.
- Peskin, C. S., 1972, Flow patterns around heart valves: a numerical method, *Journal of Computational Physics*, vol. 10, pp. 252-271.
- Piechna, J.R. and Meier, G.E.A., 1986, Numerical Investigation of Steady and Unsteady Flow in Valve Gap, *International Compressor Engineering Conference at Purdue*.
- Possamai, F. C.; Ferreira, R. T. S.; Prata, A. T., 2001, Pressure distribution in laminar radial flow through inclined disks. *International Journal of Heat Fluid Flow*, New York, vol. 22, n. 4, p. 440-449.
- Raal, J.D., 1978, Radial Source Flow Between Parallel Disks, *Journal of Fluid Mechanics*, vol. 85, p. 401-416.
- Tabatabai, M., Pollard, A., 1987, Turbulence in Radial Flow Between Parallel Disks at Medium and Low Reynolds Numbers, *Journal Fluid Mech.*, vol. 185, p. 483-502.
- Wark, C.E., Foss, J.F., 1984, Forces Caused by the Radial Out-Flow Between Parallel Disks, *Journal of Fluids Engineering*, vol. 106, p. 292-297.

ACKNOWLEDGEMENT

The authors acknowledge the support given by FAPESP and FUNDUNESP.